

Study of Numerical Method for Turbulent Flow Simulation around Complex Geometries (複雑形状 物体に対する乱流シミュレーションのための数値計 算法に関する研究)

著者	釜土 敏裕
号	53
学位授与番号	4020
URL	http://hdl.handle.net/10097/42434

氏 名	かま っち とし ひろ
授 与 学 位	釜 土 敏 裕
学位授与年月日	博士 (工学)
学位授与の根拠法規	平成20年9月11日
研究科, 専攻の名称	学位規則第4条第1項
学 位 論 文 題 目	東北大学大学院工学研究科 (博士課程) 航空宇宙工学専攻
	Study of Numerical Method for Turbulent Flow Simulation
	around Complex Geometries
	(複雑形状物体に対する乱流シミュレーションのための
	数値計算法に関する研究)
指 導 教 員	東北大学教授 中橋 和博
論 文 審 査 委 員	主査 東北大学教授 中橋 和博 東北大学教授 澤田 恵介
	東北大学教授 大林 茂 東北大学教授 山本 悟
	(情報科学研究科)

論 文 内 容 要 旨

Chapter 1: Introduction

With dramatic improvement of the recent computer performance and development of the mesh generation technology for the complex geometry, the application of Computational Fluid Dynamics (CFD) becomes large-scale, complicated and multiple. Although the automatic mesh generation is becoming possible, the time to make the calculation mesh requires several days or more generally. Moreover, the computing time of flow simulation becomes ten hours or more in CPU time by using the super computer. Based on such a current state, the problems that current CFD has are; (i) to secure of calculation accuracy for a reproduction of the experiment result, (ii) to shorten the time both of mesh generation and computation, (iii) to treat complex geometries consist of many components, and (iv) to cancel the difference of the mesh quality for the relative performance evaluation. To solve these problems, the evolution of the robust and automatic mesh generation technology and the development of the numerical method to shorten the computational time even with a more fine mesh are required.

Another request to the current CFD is the numerical aeroacoustic analysis for predicting the aerodynamic sound generated from turbulent flow. The aeroacoustics has the character to grow rapidly with the increase of the reference flow velocity. Therefore, evaluation and reduction of aeroacoustics become important problems in wide engineering fields. Predictions of the aeroacoustics were mainly based on the experiment or the theory so far. With the development of resent super computers, aeroacoustic predictions are extremely conducted with an extension of CFD or coupling CFD and the acoustic analysis. The research on the aeroacoustic prediction by the numerical calculation is called Computational AeroAcoustics (CAA), and being systematized as a relatively new field of study.

The objective of this thesis is to develop the numerical methods that are required to simulate flows around complex

geometries. The final target of this research is to realize high fidelity turbulent flow simulations of engineering applications. To achieve this goal, the numerical methods to accurately simulate aeroacoustics and high-Reynolds number flows around two and three dimensional complex geometries have been developed. Unstructured mesh and Cartesian mesh were employed as the basic approach for CFD because of the flexibility to treat complex configurations such as a whole vehicle. The computational results have been compared with other available data and the validity of the methods has been discussed.

Chapter 2: Fluid/Aeroacoustics Coupling simulation Using Unstructured Mesh Method

The term “aerodynamic sound” refers to the acoustic fields caused by air flows. Recently, aerodynamic noise has become the dominant factor in the noise emitted by automobiles, aircraft, and other vehicles as operating speeds increase. There is heightening demand from society to reduce such noise so as to preserve physical comfort and the livability of the environment. Therefore, it is of growing interest to be able to predict noise levels. Conventionally, predictions of aerodynamic noise levels have relied mainly on experiments and theory, but the recent advances in the techniques of CFD have allowed researchers to publish many numerical studies on prediction of the noise. The objective of this chapter is to develop a new computational method for predictions of aerodynamic sound fields. In the computational analysis, numerical method to predict aerodynamic sound generated from turbulent fields was discussed. The sound radiation was analyzed with Linearized Euler equations (LEE) with source terms of sound generation. LEE simulation with sound source terms is not limited by any assumptions of homogeneous media; it can also handle diffraction and wall reflection. The sound generation was computed with the Stochastic Noise Generation and Radiation (SNGR) model using local turbulence scales obtained from Reynolds Averaged Navier-Stokes (RANS) flow simulation. In the analyses, an unstructured grid method was employed for effective handling of complex geometric models. Computational results were validated with simulation of propagation of acoustic pulses and of turbulent velocity fields. This method was then applied to flow simulation around an airfoil for predictions of aeroacoustics.

Chapter 3: Turbulent Flow Simulation Using Building-Cube Method

The progress in computer performance makes it possible to solve very complex flow problems. However, the mesh generation process for a complex geometry still requires large amounts of time and labor, which makes it difficult to reduce the overall time and operation cost for solving practical problems such as flow simulation around a whole vehicle. Although the calculation time for solving the flow itself becomes shorter, the time required for grid generation cannot be reduced without automating the mesh generation process. Recently, CFD using Cartesian mesh has been revisited for easy mesh generation and robust computation. The most critical issue of the conventional Cartesian mesh, however, is how to fit the grid spacing to the local flow scale without introducing algorithm complexity. For a flow computation around a wing, for example, the grid size near the wall boundary is restricted to the smallest scale of the flow feature to be resolved. Although the Cartesian mesh with

the cut-cell approach has the capability for use with a complex geometry, this approach is basically the same as the unstructured mesh approach, and the advantages of the Cartesian mesh such as its simplicity and low memory requirement may be lost. To solve these problems and aim for next generation CFD algorithm, new Cartesian mesh approach, named Building-Cube Method (BCM) was proposed. The Building-Cube method is basically based on a block-structured Cartesian mesh approach to realize the high-density mesh computations for large-scale computation. The objective of this chapter is to propose a new CFD algorithm based on BCM with an improved boundary treatment for high-Reynolds number flows of engineering applications. The BCM allows a fast, low-storage method for time-accurate computation of the Navier-Stokes equations for a complex geometry. The simplicity in using Cartesian mesh will become important for mesh generation, the solution algorithm, and the post processing involved in a large-scale computation. In the original BCM, however, the wall boundary is defined by a staircase representation in order to keep the simplicity of the algorithm. Therefore, to resolve viscous sublayers in the boundary layers, vast amounts of grid points are required especially for high-Reynolds number flows of engineering applications. In this study, to represent an accurate curved solid boundary with reasonable mesh points, an embedded boundary method is applied at the wall boundary on the Cartesian cells: that is, source terms are added to the Navier-Stokes equations for the immersed surface. The embedded technique was originally developed for incompressible flows as an immersed boundary method, and very few applications have been presented for the computation of compressible flow so far. In this work, the embedded boundary treatment is newly extended to the compressible Navier-Stokes equations on the Cartesian mesh with gridless method. This approach allows easy mesh generation around a complex geometry and a fast, low-storage computation for high-Reynolds number flows of engineering applications. The developed algorithm was applied to many bench mark problems to evaluate the method.

Chapter 4: Flow Simulation around Complex Geometries Using Adaptive Cube Refinement

One of the advantages of Cartesian grid methods is that they allow for simple local grid clustering, which becomes an advantage especially in performing solution adaptive computation. Coupled with a tree-based data structure and grid adaptation, with respect to both the geometry and the flow field, these methods have been demonstrated to be very viable tools for flow simulation, with very complex geometry. With the adaptive Cartesian grid approach, grid-independent solutions can be obtained with automated solution-based grid adaptations. To achieve the same solutions with a non-adaptive mesh would be very expensive. One obvious drawback of the adaptive Cartesian grid method is its inability to support directional grid adaptation required in viscous boundary layer type flow problems. Isotropic grid adaptations in a boundary layer are not only too expensive but inefficient as well. Furthermore, the irregular cut cells near the solid wall boundaries have been shown to produce severely non-positive numerical stencils for the Navier-Stokes equations and may cause convergence/stability problems. In this chapter, a new adaptive mesh refinement algorithm with BCM is discussed to compute viscous flow around

3D complex geometries. One of the popular data structures for adaptive Cartesian grid is the Octree. The drawback of Octree is that only isotropic grid refinement is supported. However, in order to reduce memory storage, it is effective to introduce anisotropic cube subdivision, since isotropic cube subdivision unnecessarily increases the number of cells, for example, when applied to a boundary layer. In this study, an Omnitree data structure has been used. The Omnitree supports many types of subdivisions, and therefore allows the adaptive Cartesian grid to be refined in a non-isotropic manner. To achieve automation in flow simulation, solution based grid adaptation is essential. To take full advantage of the anisotropic grid adaptation capability offered by the Omnitree, the three coordinate directions of each Cartesian cell are examined independently for possible adaptation. The implementation of the Adaptive Cube Refinement (ACR) is straightforward in the present approach because the adaptive refinement is applied not to cells but to cubes. Since the number of cubes is relatively small, the cube refinement is quite simple and the refinement does not cause the difficulty of the dynamic load balancing in parallel computations. The domain decomposition technique is used for the parallelizing strategy. All cubes should be partitioned so that the leaf cubes are equally distributed to each processor, since only the leaf cubes are used for flow computation. In this work, Space-filling curves (SFCs) are employed for parallelization. In the scalability test of parallel computation, good performance was shown. This method efficiently captured flow features such as shock waves, wakes, or the large separation vortex. As one of the practical engineering applications for the most complex geometry, flow simulation past a race car was conducted with the developed method.

Chapter 5: Conclusions

The objective of this thesis is to develop the numerical methods that are required to simulate flows around complex geometries. The final target of this research is to realize high fidelity turbulent flows of engineering applications. To achieve this goal, the numerical methods to accurately simulate aeroacoustics and high-Reynolds number flows of 2D airfoils and 3D complex geometries have been developed. Unstructured mesh and Cartesian mesh were employed as the basic CFD approach because of the flexibility to treat complex configurations such as a whole vehicle. The computational results that have been compared with other available data showed the validity and effectiveness of the methods.

論文審査結果の要旨

計算流体力学 (CFD) は今日では航空機や自動車の空力設計において重要な役割を果たしている。しかしながら、現状の CFD は計算精度や適用限界、あるいは使い勝手等のために工学設計ツールとしては未だ不十分である。そのため、全機形状等のより複雑形状物体の取扱い能力改善や、空力騒音などを含む大規模な乱流シミュレーションに対する高精度計算が可能な手法の開発が強く求められている。本研究は、今後の計算機能力の向上を勘案して CFD の工学応用を更に高めることを目指し、複雑形状物体に対する乱流シミュレーションのための新しい数値計算法を構築することを目的としたものである。

本論文は、これらの研究成果をまとめたものであり、全編 5 章からなる。

第 1 章は導入であり、本研究の背景、目的および構成を述べている。

第 2 章では、乱流場から発生する空力音の計算手法について議論している。音の伝播計算には線形化オイラー方程式を用いている。音源はレイノルズ平均ナビエ・ストークス計算 (RANS) から得られた乱流エネルギーと散逸率から Stochastic Noise Generation and Radiation モデルを用いて計算している。流体と空力音の解析には、複雑形状への対応性から非構造格子法を採用している。計算結果は従来の構造格子による LES を用いた結果と定性的に一致することが示されている。これは、RANS を用いた空力音解析について有益な成果である。

第 3 章では、工学問題のための大規模で高精度な流体計算法について議論している。壁面セルに対して、はめ込み型境界条件を改良し、ベースの直交格子として Building-Cube Method (BCM) を適用する計算法を提案している。本手法により、複雑形状物体に対する計算格子の容易な生成、高速で低メモリの計算が可能となる。本手法の検証は低速および高速の様々な流れで行われ、その妥当性が示されている。これは、直交格子、特に BCM を工学問題へ応用する際の有効かつ非常に重要な成果である。

第 4 章では、BCM を用いた解適合格子法について議論している。分割はキューブ単位で行われ、分割の際の動的負荷バランスのアルゴリズムを単純化している。分割の際には非等方分割を採用して流れ場や物体に対する格子生成をより効率化している。実問題における計算高速化のために並列化が行われ良好な計算効率が得られることを示している。本手法の検証は衝撃波を伴う流れや離流れおよび後流等について行われ、その有効性が示されている。工学問題の実用例としてレースカーまわりの流体計算が行われ、非常に複雑な流れ場をとらえることに成功している。以上の結果は、BCM の産業界への実用化に向けた重要な成果である。

第 5 章は結論である。

以上要するに本論文は、複雑形状物体の取扱い、および大規模な乱流シミュレーションに対する高精度計算を可能とする手法について提案しているものであり、航空宇宙工学および計算流体力学の発展に寄与するところが少なくない。

よって、本論文は博士(工学)の学位論文として合格と認める。